Viewing Files with Volo View

Working with Files in Volo View
Viewing Files with an Internet Browser

This section describes how to use Volo™ View to view DWG, DXF™, or DWF files, including working with Volo View commands and setting Volo View options.

Working with Files in Volo View

You can use Volo View to view different file types including AutoCAD® drawings (DWG), drawing exchange format (DXF), and drawing web format (DWF) files. There are several ways to view a file with Volo View:

- Start Volo View, then from the Windows Explorer, drag and drop a file into Volo View.
- Start Volo View, then from the Volo View File menu, choose Open. Enter the path and name of the file to view.
- Double-click a file that is associated with Volo View in Windows Explorer. Volo View launches and displays the file.
  Note: File association depends on the applications you have installed or uninstalled. See File Associations for more information.

Viewing Files with an Internet Browser

When you install Volo View, you also install the Volo View ActiveX control. If Netscape is installed on your computer, the Volo View installation also installs a Netscape Plug-in for the ActiveX control. You can view a file with Volo View and Microsoft Internet Explorer, or with Volo View and Netscape Navigator, in the following ways:

- Launch a browser, then from Windows Explorer drag and drop a DWG, DXF, or DWF file into the browser.
- Launch a browser, then from the browser's File menu, choose Open. Load an HTML file that has a DWG, DXF, or DWF file reference. See Adding Drawing File References to Your HTML Files for more information.

Volo View Menu Commands

The Volo View commands that are available depend on how you are using Volo View. If you are using the Volo View ActiveX control, you can use a right-click menu to choose from the commands described in Right-Click Menu Commands. If you are using the Volo View application, you can use the right-click menu of commands, or you can use any of the commands in the application menu bar.

The following menus and commands are always available in the application:

- **File**
  Use the File menu commands to open and close files, manage printer settings, print the current view, open recently used files, send files as email attachments, and exit the application.
- **View**
  Use the View menu commands to control display of the application toolbar and status bar. To hide the toolbar or status bar, choose the appropriate checked command. To display the toolbar or status bar, choose the unchecked command.
  Note: When a drawing file is open, the View menu contains the options that are also available on the Right-click menu.
- **Help**
  Use the Help menu commands to access help on Volo View, or to display the About Volo View dialog box from which you can view important file information. For more information on using the About Volo View dialog box to display file information, see Getting File Information.

The following menus and commands are available in the application when a file is open:

- **Edit**
  Use the Edit menu command to copy the current view of a file. For more information on copying a view from Volo View, see Copying and Pasting the Current View.

- **Tools**
  Use the Tools menu commands to set Volo View options, display or hide scroll bars, and control the display of the view. The menu commands that control the display of the view are also available on the Volo View right-click menu.
- **Window**
  Use the Window menu commands to arrange the display of windows in the application. You can cascade or tile open files. To make a file current, select the file from the list of open files at the bottom of the menu.

Right-Click Menu Commands

To view the right-click menu of Volo View commands, from your browser or within an application, right-click in the Volo View window. Available Volo View menu commands are:

- **Pan**
  To move the drawing display in the current view choose the Pan command. To pan, click in the Volo View window, then drag. Volo View is by default in Pan mode and the mouse pointer is a hand icon. The hand icon grabs the screen as you pan. To learn more about panning the current view, see Working in Pan Mode.
  Keyboard shortcut: Arrow keys
- **Zoom**
  To reduce or increase the magnification of the current view, choose the Zoom command. When the Zoom command is active, the pointer changes to a magnifying glass. To zoom, click the left mouse button and drag up and down in the window to zoom in and out, respectively. For more information on using the Zoom command, see Working in Zoom Mode.
  Keyboard shortcut: Pg Up and Pg Dn
- **Zoom Window**
  To increase the magnification of the view to a rectangular region you specify, choose the Zoom Window command. When Zoom Window is active, the pointer becomes a cross hair. To use Zoom Window, press the left mouse button and drag a rectangle over the area you want to view. You must zoom to an area of at least 5 pixels. For more information on using the Zoom Window command, see Zooming a Specified Region.
- **Zoom Extents**
  To fit the drawing to the Volo View window, choose the Zoom Extents command. For more information on using the Zoom Extents command, see Zooming the Drawing Extents.
  Keyboard shortcut: Home
- **Orbit**
  To change the perspective of the view to a different location in 3D drawing space, choose the Orbit command. You can manipulate the perspective by clicking and dragging the mouse. When the Orbit Command is active, the target of the view stays stationary and the camera location, or point of view, moves around the target. To learn more about working with Orbit, see Orbiting the Current View.
  Note: This command is only available when viewing DWG or DXF files and is not available in Layouts.
- **Layers**
  To turn drawing layers on (shown) and off (hidden), choose the Layers command. The Layers dialog box opens. You can select one or more layers, then click the light bulb icon to turn on or off all of the selected layers. Layers are toggled based on the state of the light bulb icon when you click it. For more information on using Volo View to work with drawing layers, see Working with Layers.
- **Named Views**
  To change the view to a named view, or to change to a standard orthographic or isometric view, choose the Named Views command. A dialog box opens from which you can select a named view or a standard view. If a drawing file contains named views, you can change the current view to a named view. If a DWG file does not contain named views, you can change the current view to one of the standard orthographic or isometric views. For more information on viewing named views, see Viewing Named Views.
- **Layouts**
  To view a previously defined layout, choose the Layouts command. The Layouts dialog box opens. Double-click the layout, or select a layout and then click the Close [x] button to dismiss the dialog box. For more information on using the Layouts command, see Viewing Drawing Layouts.
  Note: The Layouts command is available only when a DWG or DXF file contains at least one layout. This command is unavailable when viewing DWF files.
- **Markup**
  To add lines and text to a drawing file, and to save the drawing file with the markups, right-click and choose Markup. A cascading menu displays the following options:
  - **Sketch:** Choose Sketch to freehand draw using the left mouse button.
  - **Comment:** Choose Comment to type text in using the keyboard.
  - **Save:** Choose Save to save a DWF file of the drawing file including any markups you entered.

  See Adding Markup Lines and Comments to Your Drawing for more information.
- **Full View**

To fill your Internet browser window with your drawing, choose the Full View command. The Full View command fills the entire browser window with your drawing, without displaying the HTML page from which the drawing is referenced.
Note: The Full View command is only available if you are working with Volo View in an Internet browser. For more information on using the Full View command, see Enlarging the View of Your Drawing.

- **Show Hyperlinks**
  To identify all the hyperlinks in the current view, choose the unchecked Show Hyperlinks command. Each hyperlink region flashes. To cancel the command, choose the checked Show Hyperlinks command from the right-click menu. To learn more about working with hyperlinks in drawings, see Viewing Drawing Hyperlinks.
  Note: The Show Hyperlinks command is available only when a drawing contains at least one hyperlink.
- **Copy**
  To copy the current view of a file, choose the Copy command. The current view is copied as an Enhanced Windows Metafile (EMF) and placed on the Windows Clipboard. For more information on copying a view from Volo View, see Copying and Pasting the Current View.
  Keyboard shortcut: CTRL+C.
- **Print**
  To print a view, right-click and choose Print. The view that is currently visible in the Volo View window is printed using the standard system print controls. To print the entire HTML page, including the current view of the drawing in the Volo View window, use the browser's Print button or File menu Print command. For more information on printing, see Printing Files.
- **Save Copy As**
  To save a drawing to your local drive, right-click and choose Save Copy As. If the file type you choose to save a copy as exists on the same server where the currently viewed drawing resides, you can save a copy of the file in the selected format locally. For example, if you are viewing a DWF file, and if a DWG file version of the current DWF file exists on the server, you can save the DWF as a DWG file. If a DWG file does not exist on the server, you cannot save a copy of that DWF drawing as a DWG file. For more information on saving copies of files, and drawing security issues, see Saving a Copy of the Drawing.
- **Options**
  To open the Options dialog box, use the Options command. From the Options dialog box you can set user paths, display options, and print options. For more information on Volo View options, see Setting Volo View Options.
- **About Volo View**
  Use the About Volo View dialog box to view information about Volo View and drawing information on the current file. To open the About Volo View dialog box, right-click and choose About Volo View. For more information on using the About Volo View dialog box to display file information, see Getting File Information.

Setting Volo View Options

You can use the Options dialog box to set user paths, display options, and print options. To access the Options dialog box, right-click in the Volo View window and choose Options. There are three tabs on the Options dialog box. Click the tab that contains the options that you want to set.

- **General:** Click this tab to specify a file name and user paths.
  - **Source Name:** Displays the name of the currently open file.
  - **User Paths:** Sets the support paths and font paths. See Setting Support and Font Paths for more information.
- **Display:** Click this tab to specify color and 3D Orbit display options.
  - **Color:** Provides options for changing the color of the background or the geometry in the current Volo View window. See Changing Displayed Drawing Colors for more information.
  - **Orbit:** Specifies the display options to use when the 3D Orbit menu command is active. For information on Shading Options see, Shading Objects in the 3D Orbit View. For information on changing the projection, see Changing the 3D Orbit Projection. Click Advanced to use hardware acceleration in the 3D Orbit view. See Using Hardware Acceleration in the 3D Orbit View for more information.
- **Print:** Click this tab to specify color print options.
  - **Color:** Provides options for changing the color of the printed background or geometry. See Controlling Print Colors with the Options Dialog Box for more information.

Adding Markup Lines and Comments to Your Drawing

With Markup you can sketch on drawings, add comments to drawings, and save a copy of the drawing file that includes the Markups you added. You can open a saved Markup file to add, delete, and edit markups.

Adding Markup geometry or text creates a layer in the drawing file named _Markup. When you save a Markup copy of the drawing file, the original drawing geometry and text is saved in a layer named Original. Every saved Markup file contains two layers, Original and _Markup. The _Markup layer is always saved even if it is turned off when you save the Markup file.

Other layers that are turned off are not saved in the Markup file.

Note: Hyperlinks are disabled when the Markup options are active. This allows you to click and draw anywhere in the drawing without activating a hyperlink. The Markup menu option is not available in the 3D Orbit view.

To use a Markup option, choose Markup from the right-click menu and then choose one of the following options:

- **Sketch:** Enables you to freehand draw using the left mouse button. See To sketch in a drawing file for more information.
- **Comment:** Enables you to type text in the drawing file. See To add comments to a drawing file for more information.
- **Save:** Saves a copy of the drawing file, including the Markups. See To save a drawing file including the Markups for more information.

The following Markup topics are also available:

- To edit existing comments
- To delete Markup geometry or text

To sketch in a drawing file

1. From the right-click menu choose Markup, and then choose Sketch.
2. Hold down the left mouse button to draw free form shapes. Release the left mouse button to end the line or shape.
   or
   Click on different points in the drawing file to draw a line between the points. Double-click to terminate the line.
   All lines appear red. There are no color, line width, or line style options.
3. Right-click and choose Pan or Zoom to end the Sketch option.

To add comments to a drawing file

1. From the right-click menu choose Markup, and then choose Comment.
2. Click in the drawing and drag to create a box that you want the text to fit into. Release the mouse button.
   When you release the mouse button the text cursor is displayed in the top corner of the text box. The text box is visible in the drawing until you exit the current text box.
3. Type the text you want to add to the drawing.
   All text is in Arial font.
4. Press the ESC key to exit the current text box.
   The Comment option is still active.
5. Click and drag the mouse to add another text box. Type a comment in the new box.
   or
   To exit the Comment option, click the right-mouse button and choose a different option from the Markup menu, or choose Pan or Zoom to exit Markup entirely.

To edit existing comments

You can edit comments while the Markup Comment option is still active, or when you are in Zoom or Pan mode.

1. Move your cursor over the text that you want to edit.
   The cursor changes to an arrow.
2. While the cursor is an arrow, double-click on the comment that you want to edit.
3. Make changes to the text. Press ESC to exit the current text box.
   or
   If you are in Zoom or Pan mode, click off of the text to exit text edit.

To delete Markup geometry or text

1. Move your cursor over the Markup geometry or text that you want to delete.
   The cursor changes to an arrow.
2. While the cursor is an arrow, click on the object to select it.
   The object that you selected turns blue.

3. Press the Delete key to delete the selected object.
   or
   Click off of the object to clear the selection.

To select multiple Markup objects, hold down the **Ctrl** key while selecting objects.

To save a drawing file including the Markups

1. From the right-click menu select Markup, and then choose Save.
   The Save dialog box appears. The File Name box contains the following file name:
   Markup_of_[filename].dwf - where [filename] is the original drawing file name.
2. Click OK to save a copy of the drawing file including the Markups.

Working in Pan Mode

In Pan mode, you can pan the drawing image to a new location. Hold down the mouse button and move the hand cursor to pan the drawing.

To pan

1. From the right-click menu, choose Pan.
2. Place the pointer over the drawing, then hold down the mouse button and move the mouse to reposition the drawing.

Note: If you are using a Microsoft IntelliMouse, you can hold down the wheel button and move the mouse. To learn more about using the IntelliMouse with Volo View, see Using an IntelliMouse with Volo View.

Working in Zoom Mode

A specific magnification, position, and orientation is known as a view. The most common way to change a view is to use the Volo View zoom commands to increase or decrease the size of the image displayed in the drawing area. Increasing the apparent size of the image to view the details more closely is called zooming in. Decreasing the apparent size of the image to see a larger portion of the drawing is called zooming out.

Zooming does not change the absolute size of the drawing; it changes the size of the view within the drawing area. Volo View provides realtime zooming.

The following zoom mode topics are available:

- Zooming a Drawing
- Zooming a Specified Region
- Zooming the Drawing Extents

Zooming a Drawing

In Zoom mode, you can zoom in or out of the drawing by moving the pointer vertically up or down. When you release the mouse button, zooming stops. You can release the mouse button, move the pointer to another location in the drawing, and then press the mouse button again and continue zooming from that location.

To zoom

1. From the right-click menu, choose Zoom.
2. To zoom in or out to different magnifications, place the pointer over the drawing, then hold down the mouse button and move the pointer vertically.

Note: If you are using a Microsoft IntelliMouse, zoom in by rotating the wheel forward and zoom out by rotating the wheel backward. To learn more about using the IntelliMouse with Volo View, see Using an IntelliMouse with Volo View.

Zooming a Specified Region

You can quickly zoom in on an area by specifying the corners that define it. The center of the area you define is the center of the new view.

To zoom in on an area by specifying its boundaries

1. From the right-click menu, choose Zoom Window.
2. Press the left mouse button and drag a rectangle over the area you want to view.
3. Release the mouse button.

Zooming the Drawing Extents

Zoom Extents displays a view that includes all objects in the drawing at the highest magnification that will fit the drawing area.

To display the drawing extents

- From the right-click menu, choose Zoom Extents.

Orbiting the Current View

To change the perspective of the view to a different location in 3D drawing space, choose the Orbit command. When the Orbit command is active, the target of the view stays stationary and the camera location, or point of view, moves around the target.

Note: The Orbit command is available only for 3D DWG and DXF drawing files, and is not available in Layouts, or DWF files.

To orbit a drawing

1. From the right-click menu, choose Orbit.
   The 3D orbit view displays an arcball, which is a circle divided into four quadrants by smaller circles.
2. Place the pointer over the drawing, then click and drag to manipulate the perspective of the view.
   When you move your cursor over different parts of the arcball, the cursor icon changes. The appearance of your cursor when you click to start dragging indicates the rotation of the view as follows:
   - When you move the cursor inside the arcball, the cursor icon changes to a small sphere encircled by two lines. By clicking inside the arcball and dragging you can manipulate the view freely. It works as if your cursor were grabbing a sphere surrounding the objects and dragging the sphere around the target point. You can drag horizontally, vertically, and diagonally.
   - When you move the cursor outside the arcball, the cursor icon changes to a circular arrow around a small sphere. Clicking outside the arcball and dragging the cursor around the arcball moves the view around an axis that extends through the center of the arcball, perpendicular to the screen. This is called a "roll."
   - When you move the cursor over one of the smaller circles on the left or right side of the arcball, the cursor changes to a horizontal ellipse around a small sphere. Clicking and dragging from either of these points rotates the view around the vertical or Y axis that extends through the center of the arcball. The Y axis is represented on the cursor by a vertical line.
   - When you move the cursor over one of the smaller circles on the top or bottom of the arcball, the cursor changes to a vertical ellipse around a small sphere. Clicking and dragging from either of these points rotates the view around the horizontal or X axis that extends through the center of the arcball. The X axis is represented on the cursor by a horizontal line.

The following Orbit topics are available:

- Shading Objects in the 3D Orbit View
- Changing the 3D Orbit Projection
- Using Hardware Acceleration in the 3D Orbit View

Note: When Volo View is in Orbit mode, many of the Volo View commands are not available and appear gray on the right-click menu. Some of the commands that aren't available in Orbit mode are Layers, Named Views, Zoom Extents, Print, and Copy. You also cannot drag and drop a drawing from Volo View into AutoCAD when Orbit mode is active. To exit Orbit mode and access the unavailable commands, choose Pan, Zoom, or Zoom Window from the right-click menu.

Shading Objects in the 3D Orbit View

You can shade objects in the 3D Orbit view to give them a more realistic 3D appearance. You change the way objects are shaded using the different shading modes. The default shading mode is Gouraud.

To change the shading mode in the 3D Orbit view

1. From the right-click menu, choose Options then click Display.
2. Under Orbit, choose one of the following options:
   - **Wireframe**
     Displays the objects in Orbit mode using lines and curves to represent the boundaries.
   - **Flat Shading**
     Shades the objects in Orbit mode between the polygon faces. This gives the objects a flatter, less smooth appearance.
   - **Gouraud Shading**
     Shades the objects in Orbit mode and smoothes the edges between polygon faces. This gives the objects a smoother, more realistic appearance.
3. Click Apply to apply the changes you made.
4. Click OK to exit the Options dialog box.

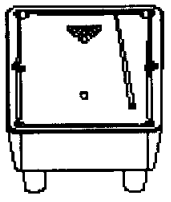
Shading is applied to objects only while in 3D Orbit mode. Switching to Pan or Zoom mode changes the current view to Wireframe with Parallel projection. If you embed Volo View software in an HTML page with the 3D Orbit view current, the drawing prints in Wireframe regardless of the current shade option.

When you orbit a large drawing file, the view may degrade to maintain the speed of the orbit. For example, if the objects are Gouraud-shaded, while you rotate the drawing, the view may degrade to wireframe or a block representation. When you stop rotating the drawing, the Gouraud shading is restored.

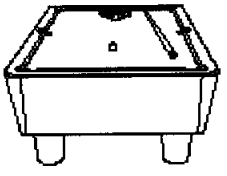
Changing the 3D Orbit Projection

When in Orbit mode, you can use a parallel projection or a perspective projection of the view:

- **Parallel:** Changes the view so that two parallel lines never converge at a single point. The shapes in the drawing always remain the same and do not appear distorted when they are closer. This is the default.
- **Perspective:** Changes the view so that all parallel lines converge at one point. Objects appear to recede into the distance while parts of the objects appear larger and closer to you. The shapes are somewhat distorted when the object is very close. This view correlates most closely to what your eye sees.



parallel projection



perspective view

The default is parallel projection, but you can change to perspective projection using the Options dialog box.

To change to the projection in 3D Orbit mode

1. While in Orbit mode, right-click in the drawing area, choose Options, then click Display.
2. Under Orbit, select Perspective or Parallel.
3. Click Apply to apply the changes you made.
4. Click OK to exit the Options dialog box.

Note: When you exit Orbit mode, projection automatically defaults to Parallel.

For more information on controlling the display of 3D graphics in Volo View, see the following topic, "Using Hardware Acceleration in the 3D Orbit View".

Using Hardware Acceleration in the 3D Orbit View

Volo View uses the Heidi® 3D graphics system developed by Autodesk as the graphics display system for Orbit mode. By default, Volo View uses the Heidi Software driver to perform all of the drawing tasks in 3D. To speed up drawing time you can use the hardware graphics card driver (hardware acceleration) to perform the drawing tasks in 3D. To use stereo viewing, you must use hardware acceleration. If you choose to use hardware acceleration, you must use the Heidi OpenGL driver.

Note: Your graphics card must support OpenGL acceleration for you to see a performance increase when using hardware acceleration.

To use hardware acceleration in Orbit mode

1. From the Volo View right-click menu, choose Options then click Display.
2. Under Orbit, click Advanced.
3. Select Use Hardware Acceleration.
4. Enter the path to the hardware driver that you want to use, or click Browse to navigate to the file. Autodesk provides the *wopenglv.hdi* hardware driver in the Volo View folder.
5. Select Stereo to set stereo viewing on.
   Note: To set stereo on you must have stereo glasses and an OpenGL accelerated video card that supports stereo viewing and stereo glasses. See your display card's vendor documentation to determine whether your display card supports stereo.
6. Click OK in the Orbit Advanced Options dialog box.
7. Click Apply and then click OK in the Options dialog box.

Enlarging the View of Your Drawing

If you are working with Volo View in an Internet browser, you can use the Full View command to fill the entire frame window with your drawing, without viewing the HTML page from which the drawing is referenced.

To fill the frame window with your drawing

- Right-click and choose Full View.

  The browser moves forward to a URL that contains just the drawing, and the drawing fills the available frame window.

To undo this operation, use your browser's Back button or Go menu Back command.

Note: The Full View command is available only when the Volo View ActiveX control is referenced from an HTML page and viewed in a browser. Full View is unavailable when using Volo View in an application environment.

For information on magnifying a view, see Working in Zoom Mode.

Changing Displayed Drawing Colors

Use Volo View options to control the background and geometry colors of the drawing you are currently viewing.

To specify background and geometry colors

1. From the Volo View right-click menu, choose Options, then choose Display.
2. To change the color of both the geometry and the background, click a button in the top portion of the Color box:
   - **Back to Default Colors:** Sets both the geometry and background colors to the color scheme that was saved in the drawing file.
   - **Set Clearscale:** Sets the geometry to grayscale and the background to white.
3. In the Settings box you can set the geometry color and the background color separately by selecting a color from the Geometry Color or Background Color list box.
   Note: If the color you want to use is not in the list, select Custom Color from the list box. The Color dialog box is displayed. Select the color that you want to use, or click Define Custom Colors to define a color. Click OK.
4. Click Apply to apply the changes you made.
5. Click OK to exit the Options dialog box.

The drawing is displayed with the color scheme you selected. Display options do not affect the printed drawing. To learn more about using Print options to control printed background and geometry colors, see Controlling Print Colors with the Options Dialog Box.

Viewing Drawing Hyperlinks

Some drawings have hyperlinks attached to drawing objects or regions. You can use Volo View to identify all hyperlinks in the current view, and to navigate to selected hyperlinks.

In Volo View, if you position the mouse pointer over a hyperlink, the pointer changes to a pointing hand icon and a tooltip displays the address (URL) of the hyperlink.

Navigating to a URL

- Click the hyperlink.
  Your browser displays the linked page. You can use the Forward and Back buttons to navigate to other web pages. However, the Back button does not return to the open drawing file in Volo View. To return to the open drawing file, close the browser window.

Showing All Drawing Hyperlinks

To identify all the hyperlinks in the current view:

- From the right-click menu, choose the unchecked Show Hyperlinks command.
  Any object to which a hyperlink is attached begins flashing.
  or
- Hold the shift key down.
  Any object to which a hyperlink is attached flashes as long as you hold the shift key down. When you release the shift key, the objects stop flashing.

Note: The Show Hyperlinks command is available only when a drawing contains at least one hyperlink.

To cancel the command:

- From the right-click menu, choose the checked Show Hyperlinks command.

Multiple URLs on a Hyperlink

If the hyperlink under the pointer contains more than one URL, the Status bar displays the message *Multiple URLs*. To display a list of URLs attached to the current hyperlink, click once. A list is displayed from which you can choose a URL.

Note that URL descriptions displayed depend on how the hyperlinks are attached to the file. In general, URLs are displayed as raw addresses, for example *http://www.autodesk.com*. However in some cases a hyperlink may contain a URL and a descriptive name, for example *http://www.autodesk.com, The Autodesk Home Page*. If a descriptive name was attached with the URL when the URL was added to the file, the descriptive name rather than the raw URL is displayed in the list of multiple URLs.

Copying and Pasting the Current View

You can copy the current view to the Windows clipboard, then paste from the Clipboard into another application, such as Microsoft Word or PowerPoint.

Note: The Copy command is not available when Orbit mode is active. To exit Orbit mode and use copy and paste, choose Pan, Zoom, or Zoom Window from the right-click menu.

To copy and paste the current view

1. From the Volo View right-click menu, choose Copy or press **CTRL+C**.

The current view is copied as a Windows Metafile (WMF) and placed on the Windows Clipboard.

2. Paste the Clipboard contents into any WMF—compliant application by choosing that application's Paste or Paste Special command, or in most cases by pressing **CTRL+V**.
   The formats and commands available depend on the application into which you are pasting the copied view. See your application documentation for more information. Refer to your Microsoft Windows documentation for more information on working with the Windows Clipboard.

Images copied and pasted from Volo View into other applications will lose some precision since WMF format files do not have the same amount of detail as DWG, DXF, and DWF files.

You can embed a drawing file in a Microsoft application, which allows users who have Volo View installed on their computer to view the drawing file using the Volo View ActiveX control. See Embedding Drawing Files in Microsoft Office Applications for more information.

Using Drag and Drop with Volo View

You can drag and drop an open file from Volo View into AutoCAD software. The type of file you are dragging and dropping, and the way that you drag and drop a file affects how the file is dropped into AutoCAD.

When you drag a DWF file from Volo View into AutoCAD software, Volo View searches the path where the DWF file resides. If the source DWG exists in the same folder as the DWF, AutoCAD uses the DWG. The source DWG file is the file that was used to create the DWF. If you want to publish a DWF, but do not want to publish the source DWG, place the DWG in a different folder than the DWF; this makes the DWG unavailable.

Note: To find the source DWG file name, choose About Volo View Express from the Help menu. The Source Filename displayed in the box at the bottom of the window is the source DWG file name.

To drag and drop a drawing file from Volo View into AutoCAD as a 2D Block

1. Start Volo View and open a drawing file.
2. Start AutoCAD and open the file to which you want to drag the open drawing file.
3. While holding down the **CTRL** key, click the left mouse button and drag the drawing file into AutoCAD software.
   The drawing is inserted into the current AutoCAD drawing as a 2D block.

To drag and drop a drawing file from Volo View and open it in AutoCAD software

1. Start Volo View and open a drawing file.
2. Start AutoCAD and open the file to which you want to drag the open drawing file.
3. While holding down the **CTRL** and **SHIFT** keys, click the left mouse button and drag the drawing file into AutoCAD software.
   The drawing file is opened in AutoCAD.

Use the Copy menu command to copy the current view to the Windows clipboard and paste it into a Microsoft Office application as a Windows metafile. See Copying and Pasting the Current View.

Note: You cannot drag and drop a drawing file from Volo View when Orbit mode is active. To exit Orbit mode and use the drag—and—drop feature, choose Pan, Zoom, or Zoom Window from the right-click menu.

Printing Drawing Files

You can use Volo View to print the current view of your drawing using the Volo View Print command. If you are using the Volo View ActiveX control in an Internet browser, you can print the current view or the entire HTML page. This section describes both printing methods and the available print options.

- Printing the Current View with the Volo View Print Command
- Printing the HTML Page with the Browser Print Command
- Controlling Print Colors with the Options Dialog Box
- Setting Printer Options with the Document Properties Dialog Box

- Printing with HP Printers

You can also print drawing files from Windows Explorer. See File Associations for more information.

Printing the Current View with the Volo View Print Command

To print only the current view of your drawing, the procedure is the same whether you are using Volo View in an application environment or as an ActiveX control embedded in an HTML page.

To print the current view

1. From the Volo View right-click menu, choose Print.
   The Microsoft Windows system Print dialog box opens. Your Windows system printer settings affect how the drawing file prints. If you are using special print settings, click Properties to verify the settings in the Document Properties dialog box. For information on using a Windows system printer, refer to your Microsoft Windows documentation.
2. Click OK to print.

Printing the HTML Page with the Browser Print Command

If you are working with a drawing that is embedded in an HTML page and you want to print the entire page, use your browser's print command. Most browsers have a Print button, or a File menu Print command that prints the entire HTML page, including the current view in Volo View. The HTML page prints according to your system printer settings. For more information on using your browser to print, refer to your browser's online help.

Note: If you embed Volo View software in an HTML page with the 3D Orbit view current, the drawing prints in Wireframe regardless of the current shade option.

If you want to print only the current view of the drawing, use the printing method described in Printing the Current View with the Volo View Print Command.

Controlling Print Colors with the Options Dialog Box

Use Volo View options to control the printed background and geometry colors for your drawing.

To specify background and geometry colors

1. From the Volo View right-click menu, choose Options, then choose Print.
   The first time the Print tab is displayed, the geometry color is set to black and the background color is set to white.
2. To change the color of both the geometry and the background, click a button in the top portion of the Color box:
   - **Back to Default Colors:** Sets both the geometry and background colors to the color scheme that was saved in the drawing file.
   - **Set Clearscale:** Sets the geometry to grayscale and the background to white.
3. In the Settings box you can set the geometry color and the background color separately by selecting a color from the Geometry Color or Background Color list box.
   Note: If the color you want to use is not in the list, select Custom Color from the Geometry Color or Background Color list box. The Color dialog box is displayed. Select the color that you want to use, or click Define Custom Colors to define a color. Click OK.
4. Select Save Print Color Settings to save the current Geometry Color and Background Color settings.
   When you select Save Print Color Settings, Volo View uses the saved colors when printing all drawing files. If you exit the application and start it again, the Print command uses the saved print color settings.
   To print an individual file using different color settings, click the Back To Default Colors button, or select different colors in the Geometry and Background Color list boxes and click Apply. This prints the current drawing using the selected colors but doesn't change the saved settings. The next drawing you print will print using the saved settings. To change the saved settings, select different colors and select Save Print Color Settings again.
5. Click Apply to apply the changes you made.
6. Click OK to exit the Options dialog box.

Print options apply to any view you print using the Volo View File menu Print command, right-click menu Print command, or using your browser's Print commands. Print options do not affect the displayed view. For information on controlling display

colors, see Changing Displayed Drawing Colors.

Note: On rare occasions, when using the Volo View control in custom applications or third-party applications, print options may not be applied.

Document Properties Dialog Box

The Document Properties dialog box changes the print options for the current printer. These settings are not preserved between print commands. Before printing, check your print settings.

To access the Document Properties dialog box

1. Choose Print from the right-click menu, then click Properties.
2. Choose the Page Setup tab to change options such as the paper size, page orientation, and duplex printing.
3. Choose the Advanced tab to change options such as graphics resolution and PostScript options.
   The available options might change depending on the printer you are using. Refer to your printer documentation and the Microsoft Windows system printer documentation for more information.
4. Click OK in the Document Properties dialog box and click OK in the Print dialog box.

Printing with HP Printers

The default for HP printers is to use vector graphics when printing. If you are using an HP printer, you can get higher fidelity results printing files using raster graphics, rather than vector graphics. You can change your printer graphics settings so that your printer uses raster graphics.

To modify the printer graphics settings:

1. From the Windows Start menu, choose Settings, then choose Printers.
2. From the displayed list of printers, select the printer whose settings you want to modify.
3. From the Printers window File menu, choose Properties.
4. Modify the printer properties as appropriate so that your printer uses raster graphics instead of vector graphics.

Printing takes slightly longer, but the resulting images look much better. For more information on modifying printer graphics settings, refer to the documentation provided with your printer.

Sending a Copy of a Drawing as an Email Attachment

You can use Volo View to send a copy of the current drawing as an email attachment using the Send command on the File menu. Your email program must be MAPI compliant to work with the Send command. The Send command is only available in the application and not from the ActiveX control.

To Send a Drawing as an Email Attachment

1. From the File menu, choose Send.
   Volo View starts your email program and creates a new message with a copy of the drawing attached to it.
   Note: The first time you use the Send command, Volo View displays a configuration dialog box so that you can configure Volo View to use your email program.
2. Fill in the required fields and type a message if you want.
3. Send the email according to your email program's procedures.
   After the email is sent, your email program closes and Volo View is active.

Saving a Copy of the Drawing

With the Save Copy As menu command, you can save a copy of the currently open drawing. When saving a copy of the currently open drawing, the following rules apply:

- You can always save a copy of the currently open drawing in Windows Metafile (WMF) format.
- You can always save a copy of the currently open drawing in the same format as the open file. For example, if the name of the currently open file is *House.DWF*, you can save a copy of the file in DWF format.

- You can save a copy of the currently open drawing file to a different format. However, a file of the format you want to use, with the same file name as the currently open drawing file must exist in the same folder as the currently open drawing file. For example, if the currently open file is http://MyServer/Drawings/House.DWG, and you want to save a copy of this file to a local drive in DWF format, the file House.DWF must exist in http://MyServer/Drawings.

To save a copy of a drawing file to your local drive

1. Right-click and choose Save Copy As.
   The Save Copy As dialog box opens. The Save As Type list box contains the available drawing file formats.
2. Specify the path where you want to save the copy, and specify the file name and format, then click Save.

Security and Saving Copies of Drawings

If you do not want to publish a DWG file, but want to publish only a DWF, do not place the DWG file in the same directory as the DWF file. In this case, DWG is not an available file format when users attempt to save a copy of the drawing.

Working with Layers

You can use Volo View to turn available drawing layers on (shown) and off (hidden).

Layers in DWG and DXF Files

For DWG and DXF files, the state of the layer when the DWG was saved in AutoCAD determines whether the drawing layer is available in Volo View software. The following explains layer availability in Volo View for DWG and DXF files depending on the state of the layer when it was saved in AutoCAD software:

- **On in AutoCAD:** The layer is visible in Volo View and available in the Layers dialog box.
- **Off in AutoCAD:** The layer is not visible in Volo View, but you can use the Layers dialog box to turn the layer on.
- **Locked in AutoCAD:** The layer is visible in Volo View and available in the Layers dialog box.
- **Frozen in AutoCAD:** The layer is not visible in Volo View and not available in the Layers dialog box. If for security reasons, you don't want to publish a layer in a DWG file, freeze that layer in AutoCAD software and save the DWG. The layer is not available in Volo View.

Layers in DWF Files

For DWF files, the state of the layer and the selected plot options when you create the DWF file determine whether the drawing layer is available in Volo View.

When you create the DWF file using the AutoCAD PLOT command, you can select or deselect the option to include layer information in the DWF file. If you do not include layer information when creating the DWF, the Layer menu command is not available in Volo View. See Specifying Additional Settings for Plotted DWF Files for instructions.

In the AutoCAD Layer Properties Manager, you can select or deselect the plot option for a layer. If a layer is not selected for plotting, it is not included in the DWF file and is not available in Volo View. In addition to the plot option, the state of the layer when you generate the DWF file in AutoCAD affects the layer's availability in Volo View in the following way:

- **On in AutoCAD:** The layer is visible in Volo View and available in the Layers dialog box.
- **Off in AutoCAD:** The layer is not visible in Volo View and is not available in the Layers dialog box.
- **Locked in AutoCAD:** The layer is visible in Volo View and available in the Layers dialog box.
- **Frozen in AutoCAD:** The layer is not visible in Volo View and not available in the Layers dialog box. If for security reasons, you don't want to publish a layer in a DWF file, freeze that layer in AutoCAD software and generate the DWF. The layer is not available in Volo View.

For more information on generating DWF files, see Generating DWF Files.

To turn drawing layers on or off

1. Right-click in the Volo View window, then choose Layers.

The Layers dialog box displays all of the available layer names. If the entire layer name does not display in the dialog box, hold your cursor over the name and a tooltip displays the full layer name. The Layers dialog box works like most Explorer dialog boxes in that you can select one or more layers by simultaneously pressing **CTRL** or **SHIFT** when selecting layers.

2. Click the light bulb icon to turn on or off all of the selected layers.
   Layers are toggled based on the state of the light bulb icon when you click it.
3. Click the Close [x] button to dismiss the dialog box.

Layers Dialog Box Right-Click Menu Commands

Right-click inside the Layers dialog box to access the Layers right-click menu.

The Layers dialog box commands are:

- **On** and **Off**
  To show or hide all selected layers, choose On or Off.
- **Toggle**
  To toggle the state of all currently selected layers, choose Toggle. Any layers that are On are toggled to Off, and any layers that are Off are toggled to On.
- **Select All**
  To select all drawing layers, choose Select All.
- **Clear All**
  To deselect all selected layers, choose Clear All.

Viewing Named Views

You can change the current view to a named view or a standard orthographic or isometric view using the Named Views command.

Named Views in DWG and DXF Files

When you save a DWG file that contains named views in AutoCAD software, all of the named views are available for viewing in Volo View. If you save the DWG file as a DXF file in AutoCAD, the named views are saved in the DXF file and are also available in Volo View.

The standard orthographic and isometric views are displayed in the bottom half of the Named Views dialog box. You can select one of these views only when viewing a DWG or DXF file in model space. The standard views are not available in paper space.

Named Views in DWF Files

If a DWF file was generated from a DWG file containing named views, the DWF file contains the named views with the following limitations:

- Only the named views specific to the UCS that is current when the DWF is created are written to the DWF file. Any named views specific to other UCS orientations are excluded from the DWF file.
- If you plot a DWF in model space, only model space named views are written to the DWF file.
- If you plot a DWF in paper space, only paper space named views are written to the DWF file.
- If a named view falls outside the plotted extents of a DWF, it isn't included in the DWF file.
- If a named view is partially clipped by the plotted extents of a DWF, you can only view the unclipped portion in the DWF file.

To display a named view

1. Right-click and choose Named Views.
   A dialog box opens from which you can select a named view. If you are viewing a DWG or DXF file in model space, you can select a standard orthographic or isometric view.
   If the entire view name does not display in the dialog box, hold your cursor over the name and a tooltip displays the full view name.

2. Click the named view that you want to use.
3. Click the Close [x] button to dismiss the dialog box.

Viewing Drawing Layouts

Drawings created with AutoCAD 2000 may have layouts. A layout is a paper space environment that simulates a sheet of paper and provides a predictable plotting setup. Drawings may have multiple layouts to display various views, each of which can contain different plot scales and paper sizes. Each layout displays the drawing as it will be plotted on the sheet of paper.

To view a previously defined layout

1. Choose the right-click menu Layouts command. The Layouts dialog box opens.
2. Click the layout that you want to view.
3. To dismiss the dialog box, click the Close [x] button.

To switch from a layout back to model space, choose Model Space from the Layouts dialog box.

Note: The Layouts command is available only when a DWG or DXF file contains at least one layout. This command is unavailable when viewing DWF files.

Setting Support and Font Paths

A support path designates the location where external references reside. Likewise, a font path designates the location of drawing-specific font files. You can set multiple support and font paths by adding the paths to the Support Paths and Font Paths list boxes in the Options dialog box. Volo View searches the paths in the order they are listed.

To set support and font paths

1. From the Volo View right-click menu, choose Options, then click General.
2. To specify a support path for external references, under Support Paths, enter the desired path or click Browse to navigate to the path.
3. Click Add to add the path to the list.
4. To specify a path where Volo View can find necessary fonts for your drawings, under Font Paths, enter the desired path or click Browse to navigate to the path.
5. Click Add to add the path to the list.
6. Click Apply to apply the changes you made. Click OK to close the Options dialog box.
   Volo View saves the list of support and font paths.

   Note: You must exit Volo View and start it again to make the font path that you added available. Volo View does not use the font path you added until you exit Volo View and start it again.

To delete support and font paths

1. From the Volo View right-click menu, choose Options, then click General.
2. To delete a support path, select the path you want to delete from the Support Paths list and click Delete.
3. To delete a font path, select the path you want to delete from the Fonts Paths list and click Delete.

Getting File Information

Drawing properties contain file information for the currently open file. You enter most of the properties in AutoCAD software using the AutoCAD DWGPROPS command. AutoCAD generates other properties automatically, for example when you generate a DWF file. The information available depends on the currently viewed file type.

DWG and DXF file information

To view file information

- From the right-click menu, choose About Volo View.
  The About Volo View dialog box displays the file information.

The following DWG and DXF file properties may appear in the About Volo View box, depending on what the user entered in AutoCAD software. Other custom properties may appear in addition to these:

- **Title:** the name of the drawing file. The title can be different from the drawing file name.
- **Subject:** the subject of the drawing. Users sometimes use the subject name to group related drawings.
- **Author:** the user who created the drawing file.
- **Comments:** the general information about the drawing file.
- **Keywords:** the keywords used in AutoCAD® DesignCenter™ to locate the drawing file.
- **Last Saved By:** the name of the last user to save the drawing file.
- **Current View:** the X, Y coordinates of the bottom left and top right corners of the drawing file.

DWF File Information

The following DWF file properties may appear in the About Volo View box; other properties may appear in addition to these:

- **Author:** the user who created the drawing file.
- **Creator:** the name of the software used to create the DWF.
- **Creation Time:** the date and time that the file was created.
- **Creation GUID:** the unique identifier assigned when the file was created.
- **Modification Time:** the date and time that the file was created.
- **Modification GUID:** the unique identifier assigned when the file is modified.
- **Source Filename:** the name of the drawing file used to create the DWF file. This is the file used when executing a drag and drop of a DWF file, see Using Drag and Drop with Volo View.
- **Source Creation Time:** the date and time that the source drawing file was created.
- **Source Creation GUID:** a unique identifier for the source drawing file.

Using an IntelliMouse with Volo View

Volo View supports the Microsoft IntelliMouse. The left and right mouse buttons function the same as on a regular mouse. The additional IntelliMouse wheel and wheel button function is described below.

- Rolling the wheel is equivalent to a Zoom command.
  Rolling forward zooms in and rolling backward zooms out. Note that when zooming with the wheel, Volo View uses the cursor position as an anchor point for the zoom. As a result, if you don't move the cursor, rolling the mouse forward and then rolling the mouse backward are not reciprocal actions.
- Holding down the wheel, referred to as the wheel button, switches Volo View to Pan mode. When the mouse is stationary, the cursor displays an origin pointer. Once you start dragging the mouse, the cursor indicates the pan direction; the rate of pan is controlled by the distance that you drag the mouse from the origin point. To stop panning, release the wheel.

Note: Volo View software does not support the IntelliMouse on Microsoft Windows 95.

File Associations

Associating a file to an application causes that application to start when you open the file in Windows Explorer. To open a file in Windows Explorer, double-click the file or select the file, click the right-mouse button and choose Open. File associations also affect the Print option on the Windows Explorer right-click menu.

Installing and uninstalling Volo View or AutoCAD software affects file associations in the following way:

- **Install Volo View only:** DWG, DWF, and DXF files open and print with Volo View.
- **Install Volo View and AutoCAD:** DWF files open and print with Volo View. DWG and DXF files open with AutoCAD.
- **Uninstall AutoCAD, leaving Volo View:** Removes file associations. To re-establish the file associations with Volo View, reinstall Volo View.
- **Uninstall Volo View, leaving AutoCAD:** Temporarily removes file associations. Start AutoCAD software to re-establish the file associations.

The Volo View installation adds View with Volo View and Print with Volo View options to the Windows Explorer right -click menu for DWG, DWF, and DXF files. In this way, if you install AutoCAD software you can still use the Windows Explorer right-click menu option to view or print a DWG file with Volo View.

For more information on file associations in Windows Explorer, see your Windows documentation.

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